

AD-A057 789

TEXAS A AND M UNIV COLLEGE STATION DEPT OF AEROSPACE--ETC F/G 20/11
STATUS REPORT OF AGGIE I COMPUTER PROGRAM.(U)
APR 78 W E HAISLER

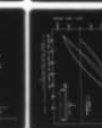
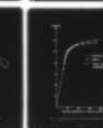
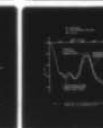
N00014-76-C-0150

UNCLASSIFIED

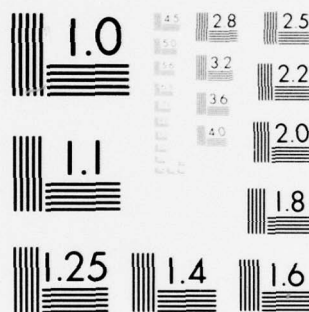
TR-3275-78-2

NL

| OF |
AD
A057789



END
DATE
FILMED
10-78
DDC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

ADA 057789

AD No. _____
UDC FILE COPY,

DISTRIBUTION STATEMENT A

Approved for public release;
Distribution Unlimited

(4)

STATUS REPORT OF AGGIE I COMPUTER PROGRAM

LEVEL II



aerospace engineering department

TEXAS A&M UNIVERSITY

WALTER E. HAISLER

DDC
RECEIVED
AUG 16 1978
B

OFFICE OF NAVAL RESEARCH
DEPARTMENT OF THE NAVY
WASHINGTON, D.C. 20025

CONTRACT No. N00014-76-C-0150
TASK No. NR 064-534

TECHNICAL REPORT No. 3275-78-2

APRIL 1978 78 07 24 065

TEXAS ENGINEERING EXPERIMENT STATION

6
STATUS REPORT OF AGGIE I COMPUTER PROGRAM.

10 Walter E. Haisler

Aerospace Engineering Department
Texas A&M University
College Station, Texas 77843

Office of Naval Research
Department of the Navy
Washington, D.C. 20025

15
Contract No. N00014-76-C-0150
Task No. NR 064-534

14
9 Technical Report, No. 3275-78-2

11 April 1978

D D C
RECEIVED
AUG 16 1978
B

DISTRIBUTION STATEMENT A

Approved for public release;
Distribution Unlimited

78 07 24 065

404 277

LB

ABSTRACT

A status report of the AGGIE I structural analysis program is presented. AGGIE I is a computer program for predicting the linear and nonlinear, static and dynamic structural response of two- and three-dimensional continuum solids. The program is based on isoparametric finite elements and allows for two-dimensional plane stress, plane strain and axisymmetric analyses and general three-dimensional analyses. Large strain kinematics is based on the total Lagrangian formulation. Materially nonlinear models include several elastic-plastic, work-hardening models as well as an incompressible Mooney-Rivlin model.

Included in this status report are several structural problems which have been investigated to verify the program. A listing of current program users is given. Also included is a summary of ongoing work which, when completed, will result in several improvements to the AGGIE I program.

ACCESSION	
NTIS	Page Section <input checked="" type="checkbox"/>
DOC	Full Section <input type="checkbox"/>
UNCLASSIFIED	<input type="checkbox"/>
JUSTIFICATION	
BY	
DISTRIBUTION/AVAILABILITY CODES	
Dist.	AVAIL. or SPECIAL
A	

SUMMARY OF AGGIE I COMPUTER PROGRAM

The purpose of this section is to provide a brief summary of the computer program as it is currently being made available to other users, the theoretical basis of the computer program, the elements used, the available material models, and the program organization. Theoretical details of the formulation may be found in Refs. 1-4 while program input instructions are given in Ref. 5.

Introduction

In beginning the development of a nonlinear structural analysis program, a review of existing nonlinear finite element programs was undertaken with the purpose being to determine what analysis capability currently existed, what techniques were being used, and to determine if any existing computer program might be suitable as a beginning base program. From this review, it was felt that the NONSAP¹ program developed by Bathe, Wilson and co-workers would form a suitable base from which to begin the work. The NONSAP program was modularized making it easy to modify, the program organization and logic was good, it was fairly well documented and it has a good library of material models and elements to conduct geometrically (large strain) and materially nonlinear analyses of two-dimensional structures (the 3-D capability was limited to linear elastic problems only).

Using the NONSAP program as a base, a moderately sized finite element program for nonlinear structural analysis has been developed. The program is based on two- and three-dimensional isoparametric solid elements. Nonlinearities accounted for in the program may be due to large displacements, large strains and nonlinear material behavior

(linear behavior is also allowed). A number of material models have been incorporated; these include linear elastic, orthotropic, nonlinear elastic, nonlinear elastic incompressible, and elastic-plastic models. Both static and dynamic response prediction are possible. The program also allows for the possibility of displacement incrementation for use in predicting buckling and post-buckling response. Dynamic dimensioning of most FORTRAN arrays allows efficient utilization of available high-speed memory. The program uses either an out-of-core or an in-core equation solver depending upon problem size and available main memory. The mesh description (node and element information) is input on a node by node and element by element basis or may be generated by using a rudimentary substructure option. Provision is made for both conservative and nonconservative (deformation dependent) loading.

Incremental Equilibrium Equations

The linearized incremental equilibrium equations which are solved at each time or load step are given by

$$[M]\{\ddot{u}^{t+\Delta t}\} + [C]\{\dot{u}^{t+\Delta t}\} + [K^t]\{\Delta u\} = \{R^{t+\Delta t}\} - \{F^t\} \quad (1)$$

where

$[M]$ = mass matrix

$[C]$ = damping matrix

$[K^t]$ = tangent stiffness matrix at time t

$\{R^{t+\Delta t}\}$ = external nodal force vector at time $t + \Delta t$

$\{F^t\}$ = nodal force vector due to internal element stresses at time t

$\{\ddot{u}^{t+\Delta t}\}$ = nodal acceleration vector at time $t + \Delta t$

$\{\Delta u\}$ = nodal displacement increment vector defined by $u^{t+\Delta t} = u^t + \Delta u$

For a static problem, the mass and damping matrices are omitted and the time increment Δt may be interpreted as an equivalent load increment.

The incremental equations given above yield only an approximate solution (because of the linearization used in their derivation) and hence will yield solutions which have accumulated error after several steps. To ensure that equilibrium is being satisfied at each step, one may use a modified Newton-Raphson method to perform equilibrium iteration. The iterative equations take the form

$$[M]\{\ddot{u}^{t+\Delta t}\}^{(i)} + [C]\{\dot{u}^{t+\Delta t}\}^{(i)} + [K^t]\{\Delta\Delta u\}^{(i)} = \{R^{t+\Delta t}\}^{(i-1)} - \{F^{t+\Delta t}\}^{(i-1)} \quad (2)$$

where (i) denotes the i^{th} equilibrium iteration and $\{\Delta\Delta u\}^{(i)}$ denotes the correction to the displacement increment at the iteration i , that is,

$$\{\Delta u\}^{(i+1)} = \{\Delta u\}^{(i)} + \{\Delta\Delta u\}^{(i)} \quad (3)$$

$$\{u^{t+\Delta t}\}^{(i+1)} = \{u^t\} + \{\Delta u\}^{(i+1)} \quad (4)$$

It should be pointed out that in a general Newton type iteration, the tangent stiffness matrix $[K^t]$ would be updated at each iteration (i.e., use $[K^{t+\Delta t}]^{(i)}$); however, in the modified Newton method, this term is held constant during the iteration (or updated only periodically when convergence deteriorates). Input parameters allow the program user to specify the updating frequency.

The time integration procedure used in dynamic analyses is either the Wilson θ or the Newmark β methods.

The original NONSAP program utilized both the total Lagrangian and updated Lagrangian formulations in developing the incremental equilibrium equations. All element and material models developed during the current research program have been restricted to the total Lagrangian

formulation; however, those material models in the original NONSAP program which premitted the use of the updated Lagrangian solution have been retained and are still available in AGGIE I.

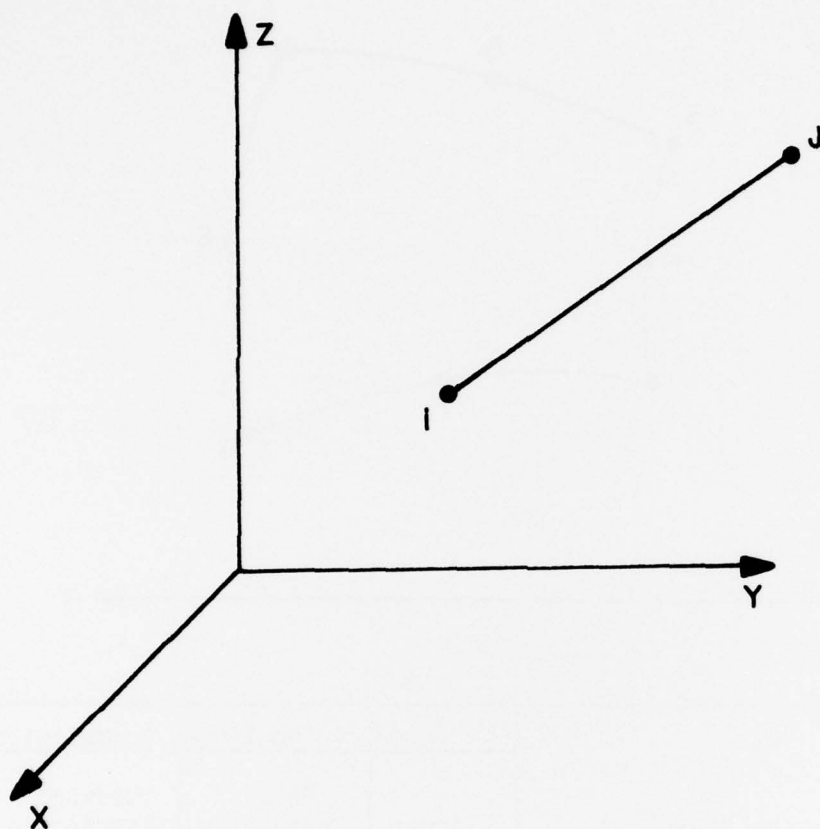
Element Library

Several types of elements are currently available in the AGGIE I program. These include a three-dimensional truss element, two-dimensional isoparametric solid element with 4 to 8 nodes, and a three-dimensional isoparametric solid element with 8 to 21 nodes. The two-dimensional element is reducable to a constant strain triangular element and the three-dimensional element is reducable to a constant strain tetrahedron.

The truss element is shown in Fig. 1 and has in general three degrees-of-freedom per node. Material properties and cross-sectional area are assumed to be constant over the length of any element. The element may be used in linear elastic or nonlinear elastic analyses in combination with small or large displacements (for a large displacement analysis, only the updated Lagrangian formulation is available).

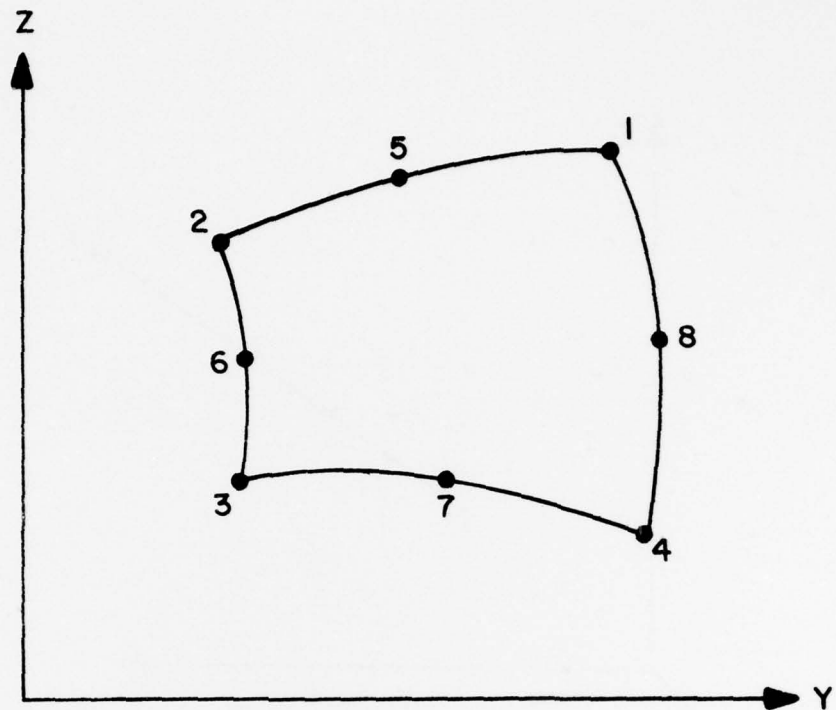
The general two-dimensional isoparametric solid element is shown in Fig. 2. The element may be used for plane stress, plane strain or axisymmetric analyses. The material models which are available for the two-dimensional element are shown in Fig. 2 and are described briefly in the next section. This element may be used to model axisymmetric shells; however, because of the continuum formulation used, the element does not represent Kirchhoff-type, thin shell behavior very well.

The general three-dimensional isoparametric solid element is shown in Fig. 3. The element may have from 8 to 21 nodes with 3 degrees-of-freedom per node. The material models which are currently available for the three-dimensional element are also listed in Fig. 3.



Available Material Models	Geometric Nonlinear Formulations		
	Linear	Total Lagrangian	Updated Lagrangian
1. linear elastic	X		X
2. nonlinear elastic	X		X

Fig. 1 Three-Dimensional Truss Element



Available Material Models	Geometric Nonlinear Formulation		
	Linear	Total Lagrangian	Updated Lagrangian
1. linear elastic, isotropic	X	X	X
2. linear elastic, orthotropic	X	X	
3. variable tangent moduli model	X	X	
4. curve description model (plane strain, axisymmetric)	X		
5. curve description model with tension cutoff	X		
6. elastic plastic, combined kinematic-isotropic hardening, von-Mises yield condition	X	X	
7. elastic-plastic, Drucker-Prager yield condition	X		
8. incompressible, nonlinear elastic, Mooney-Rivlin material (plane stress)	X	X	
9. incompressible, nonlinear elastic, Mooney-Rivlin material (plane strain or axisymmetric)	X	X	
10. elastic-plastic, mechanical sublayer model, von-Mises yield condition	X	X	

Fig. 2 Two-Dimensional Isoparametric Continuum Element for Plane Stress, Plane Strain or Axisymmetric Analysis

Material Models

The original NONSAP program contained a number of material models for the two-dimensional elements but contained only a linear and non-linear elastic material model for the three-dimensional elements. In the AGGIE program, new material models have been added for the two-dimensional elements and four material models have been added for the three-dimensional elements. Table 1 gives a complete listing of the available material models.

For the truss elements, two models are available: 1) linear elastic material and 2) nonlinear elastic material. The nonlinear elastic-material model makes use of a user-specified, piece-wise linear stress-strain curve.

For the two-dimensional solid elements, a number of models are available as shown in Table 1. Most of the material models may be used with either a geometrically linear (small strain) or nonlinear (large strain) formulation.

Two linear material models are available, i.e., either isotropic or orthotropic.

A newly developed nonlinear elastic (hyperelastic) incompressible model which makes use of Mooney-Rivlin material constants to describe the nonlinear stress-strain relation may be used for the analysis of some rubber-like materials. The plane-strain and axisymmetric models make use of Lagrange multiplier constraints in order to impose the incompressibility condition.⁴

Two new elastic-plastic material models provide a wide range of hardening rules which may be used in elastic-plastic analyses. Both models are based on the von Mises yield condition and the associated flow rule. For the hardening rule, one model makes use of a combined

Table 1. Material Models

Model	1-D	2-D	3-D
1	linear elastic	linear elastic, isotropic	linear elastic, isotropic
2	nonlinear elastic	linear elastic, orthotropic	linear elastic, orthotropic
3		variable tangent moduli model	curve description model
4		curve description model (plane strain, axisymmetric)	elastic-plastic, combined kinematic-isotropic hardening, VonMises yield condition
5		curve description model with tension cutoff	elastic-plastic, mechanical sub-layer model, Von-Mises yield condition
6		elastic-plastic, combined kinematic-isotropic hardening, Von Mises yield condition	incompressible nonlinear elastic, Mooney-Rivlin material
7		elastic-plastic, Drucker-Prager yield condition	
8		incompressible, nonlinear elastic, Mooney-Rivlin material (plane stress)	
9		incompressible, nonlinear elastic Mooney-Rivlin material (plane strain or axisymmetric)	
10		elastic-plastic, mechanical sublayer model, Von Mises yield condition	

kinematic-isotropic hardening model while the second is based on a mechanical sublayer (Besseling) model. The combined kinematic-isotropic hardening model allows for the input of a yield surface size vs. equivalent uniaxial plastic strain curve if such data is available; otherwise this information is generated by the program for a user-specified value of the ratio of kinematic-to-isotropic hardening. Details of both models may be found in Ref. 3.

Three geological material models originally in NONSAP have been retained in the present program. These are the variable tangent moduli model, the curve description model and the curve description model with tension cutoff.² The variable tangent moduli model describes an isotropic material in which the bulk and shear moduli are functions of the stress and strain invariants and the functional relationship used replaces an explicit yield condition. In the curve description model, the instantaneous bulk and shear moduli are defined by piecewise linear functions of the current volume strain. With the tension cutoff model, tensile stresses due to applied loading cannot exceed the gravity in-situ pressure; the model assumes reduced stiffness in the direction of the tensile stress which exceed in magnitude the gravity pressure (i.e., when a "crack" develops).

The material models available for the three-dimensional solid elements are also shown in Table 1. Several new models were developed and implemented in the base program. These include the Mooney-Rivlin incompressible model, the linear orthotropic model, the elastic-plastic model with combined kinematic-isotropic hardening, and the elastic-plastic model with mechanical sublayer hardening. Each of these models may be used in either a geometrically linear analysis or a large strain analysis based on the total Lagrangian formulation.

Applied Loads

The applied loads considered by the program are concentrated loads at the nodes, uniform pressure applied to the surface of an element, or uniform pressure applied to the surface of a group of elements. If a pressure is specified, the program automatically computes the equivalent generalized nodal forces.

The applied loads may be either conservative or nonconservative. Nonconservative loads are those loads which are a function of the deformed geometry, e.g., following loads or during large deformation when the surface area over which the pressure acts changes considerably.

Program Organization

The general program organization which was utilized in the base program NONSAP¹ has been retained in the present program. The significant differences are in the way the element matrices are assembled into the structural matrices and the allocation of the high-speed storage.

As in NONSAP, the solution process is divided into three phases:

1. Input and Mesh Generation Phase. Various control information is first read from data. Depending upon which input option is selected, the node and element information is either read from data or generated internally and then stored on tape or disk. Additionally, equation numbers for all active degrees are generated and element-node connectivity arrays are determined and stored on disk. Externally applied pressure loads are converted to nodal forces and stored on disk.

2. Assemblage of Constant Structural Matrices. The stiffness, mass and damping matrices for all linear elements are computed, assembled and stored on disk.

3. Step-by-Step Solution. The incremental equations of equilibrium are solved for each step. At each step, equilibrium iteration is performed (if desired) until a converged solution is obtained. Displacements, velocities, accelerations, stresses and strains are computed and printed.

The program is designed to process elements by groups; all elements within an element group have like characteristics, i.e., they are all the same element type (e.g., 2-D axisymmetric), they all have the same type of geometric nonlinearity and the same material model. Each element group generally calls for a specific overlay segment of the program. Program efficiency is increased by grouping elements to avoid unnecessary or repeated calling of overlay segments. Element groups may be either linear (geometrically and materially linear) or nonlinear. During all phases linear element groups are processed first. Stiffness properties for all linear element groups are computed during phase 2 and stored on disk. During phase 3, the structural stiffness for all linear element groups is read from disk; then stiffness matrices for all nonlinear element groups are computed (based on the current stress strain-state) and added to the linear contribution.

The program is designed to make maximum use of available high speed memory. A common block/AMAT/dimensioned for MTOT locations is used to store the arrays that are required during the different phases of program execution. During the input phase, space (NUMEST locations) is allocated in the common block for storage of element property data (element coordinates, element-node connectivity, material properties and working arrays for element stresses, strains and various variables associated with nonlinear material models). For each element group, the input is processed using the same NUMEST locations and necessary information is written on disk. During the solution phase, the element

property data is read from disk and the same space is used repeatedly as each element group is processed. The program automatically allocates the NUMEST locations within the common block based on the storage requirements of the largest element group. During the input phase, the program determines the storage that will be required for the structural coefficient (e.g., stiffness in static analyses) matrix and the space (LBLOCK) that will be available with the common block. If sufficient space is available in the common block to store the coefficient matrix in core, then this is done. If sufficient space is not available, then the coefficient matrix is stored on disk in blocks of length LBLOCK and then processed block-by-block within the LBLOCK locations of common block AMAT.

The current version of the AGGIE program contains over 125 subroutines and is overlaid (See Table 2). Descriptions of the subroutines and the overlay structure are given in the Programmer's Guide.

Equation Solvers

The solution of the incremental equilibrium equations is obtained using either the in-core linear equation solver OPTSOL⁶ or the out-of-core linear equation solver OPTBLK⁷. The coefficient matrix (left side of the equations) is stored as a one-dimensional array using a skyline storage scheme (by columns) i.e., only the terms in each column which fall below the skyline are stored. If the coefficient matrix will fit within the high-speed memory space which has been allocated to it, then the solution of the equations is done in-core by OPTSOL. If the coefficient matrix will not fit entirely in-core, then it is stored on disk, tape or other low-speed storage device by blocks and solved by OPTBLK in a block-by-block fashion. The blocksize is selected as large as possible depending upon available high-speed memory.

VERIFICATION PROBLEMS

The results of three structural problems which have been used to verify the AGGIE program are presented here. Although the geometries considered here are rather simple, more complex geometries requiring about 2,500 degrees-of-freedom have been studied. Details of the geometry, loading and material properties are given in Ref. 5.

Figure 4 shows a thin, shallow, spherical shell which is subjected to a uniform step pressure. Figure 5 presents a comparison of the apex deflection obtained with the AGGIE program (ten 8-node axisymmetric solid elements), the DYNAPLAS thin shell program (20 thin, axisymmetric shell elements) and a pseudo force solution (a modified DYNAPLAS program). The agreement between the solutions is quite good.

Figure 6 depicts an axisymmetric steel pressure vessel subjected to a static internal pressure. Results of the AGGIE program using 45 isoparametric elements are compared in Fig. 7 to another finite element solution obtained by Zienkiewicz. The solutions, which assumed the material was elastic-perfectly plastic, agree reasonably well.

Figure 8 shows a simply supported circular plate of high strain-hardening 2024-T3 aluminum subjected to a cyclic punch load at the center of the plate. The elastic-plastic behavior was modeled with the combined kinematic-isotropic hardening material model (ratio of isotropic to kinematic, $\gamma = 0.5$). The AGGIE results are compared to experimental data provided by Grumman Aerospace Corp. Results for this problem obtained with other elastic-plastic material models are reported in Refs. 3 and 5.

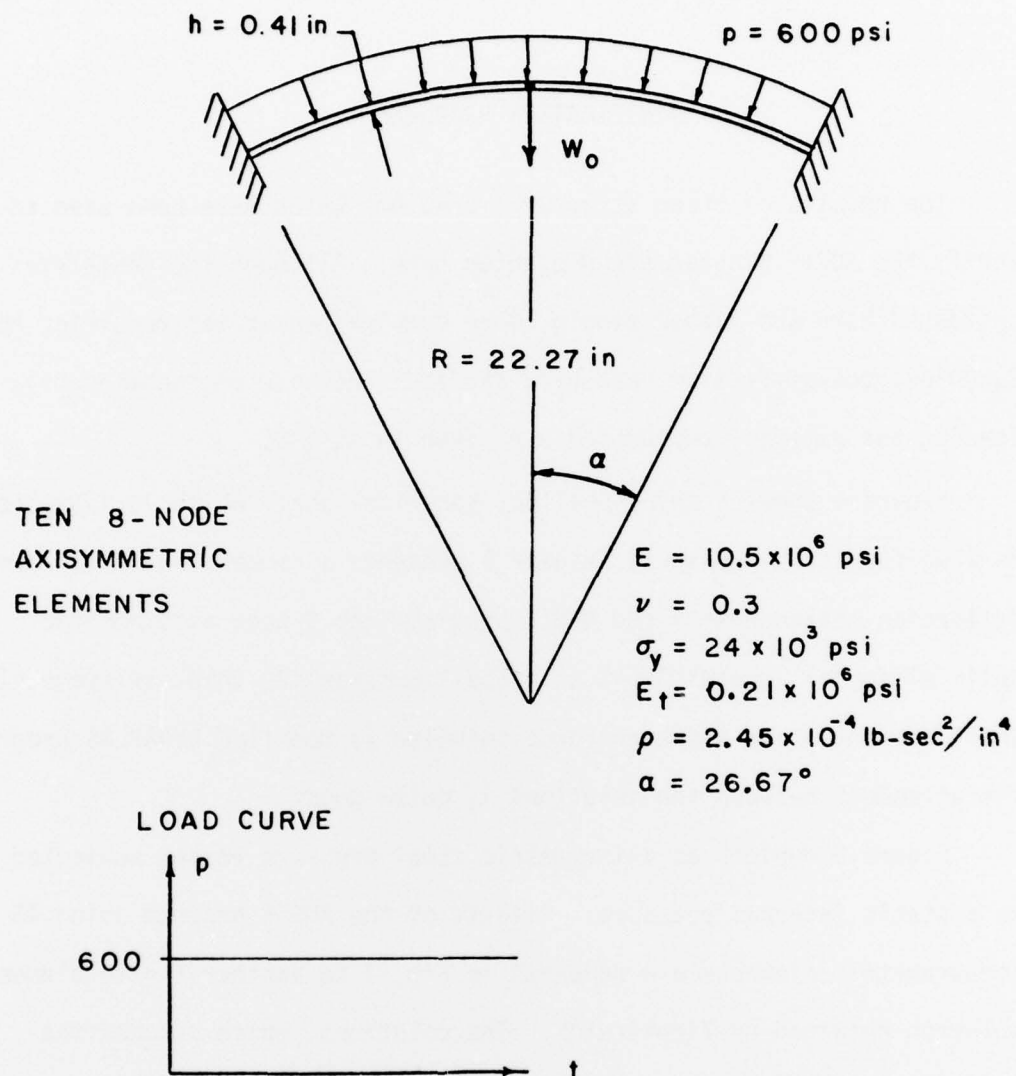


Fig. 4. Spherical cap with a uniform pressure loading.

ALL SOLUTIONS:
T.L. WITH MECHANICAL SUBLAYER
 $\Delta t = 5 \mu \text{ sec.}$

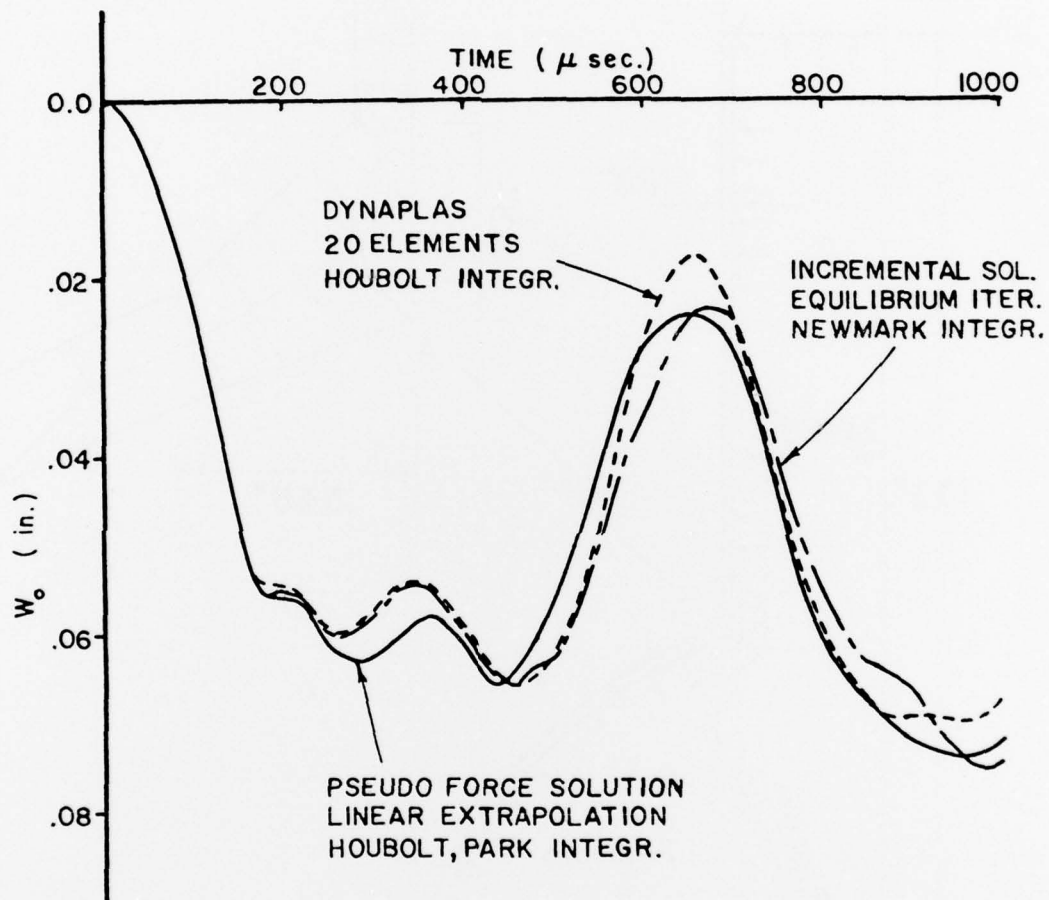


Fig. 5 . Comparison of different nonlinear program solutions for a spherical cap.

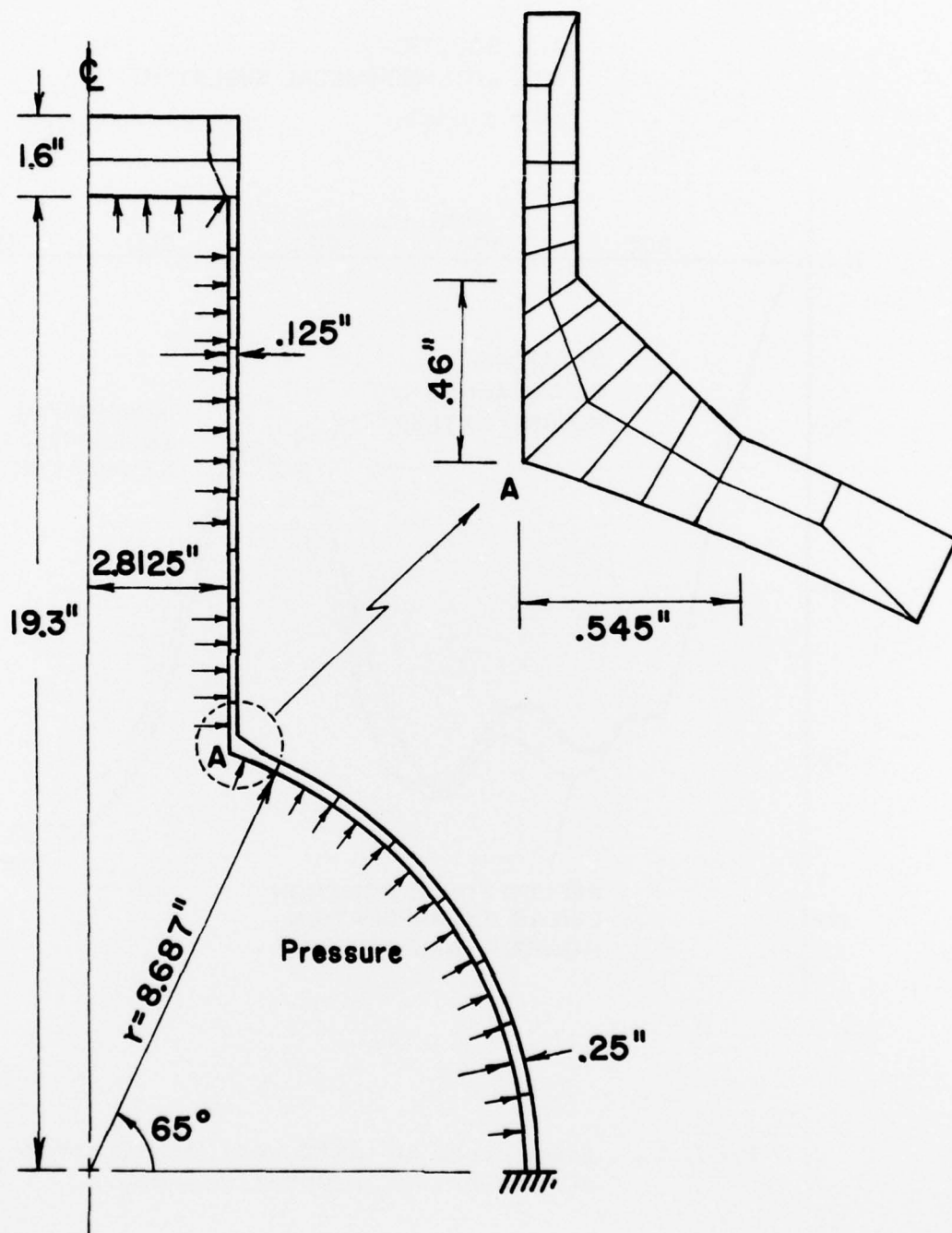


Fig. 6. Axisymmetric Steel Pressure Vessel Subjected to Uniform Internal Pressure

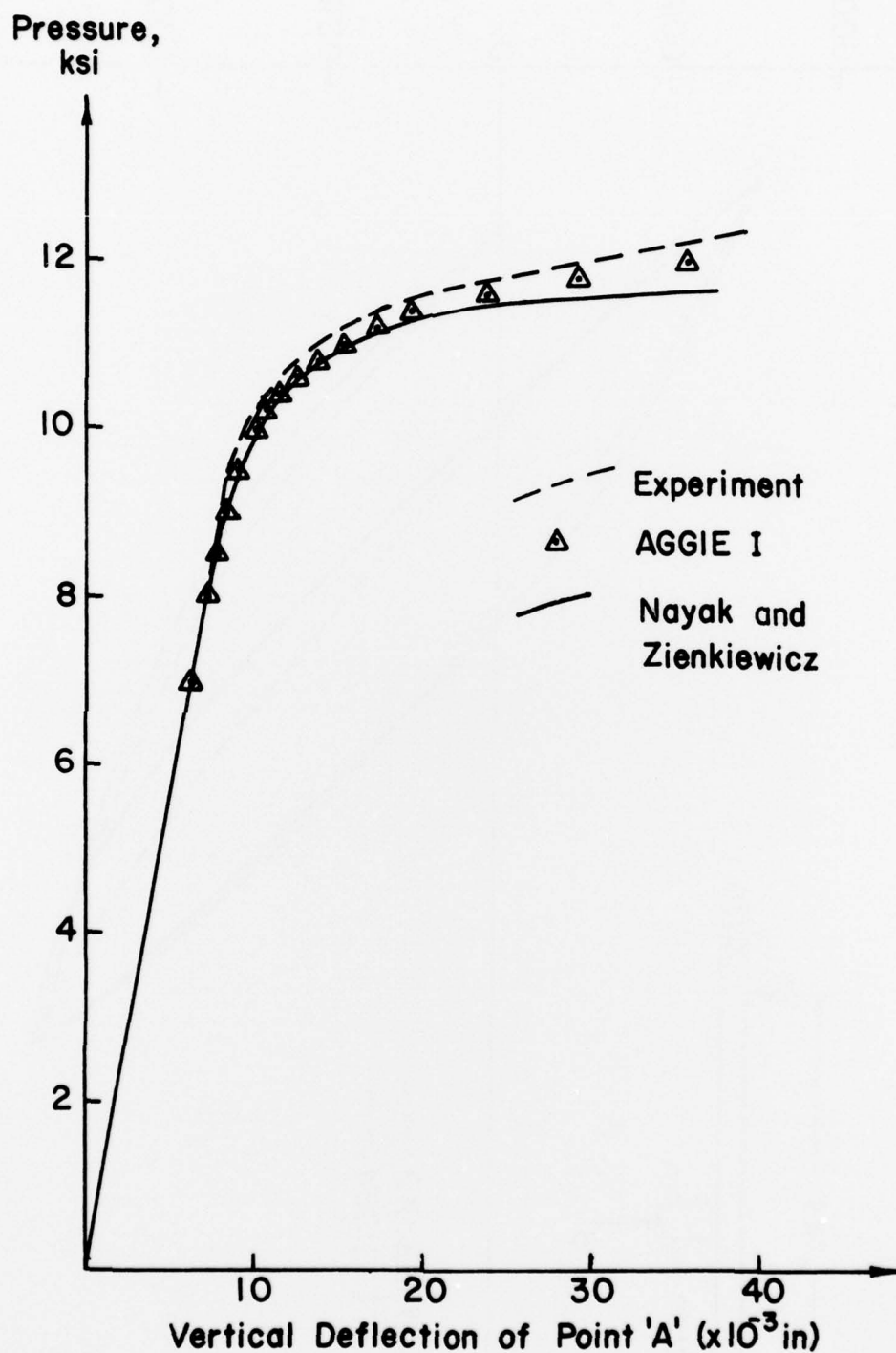


Fig. 7. Vertical Deflection of Point A for Axisymmetric Steel Pressure Vessel

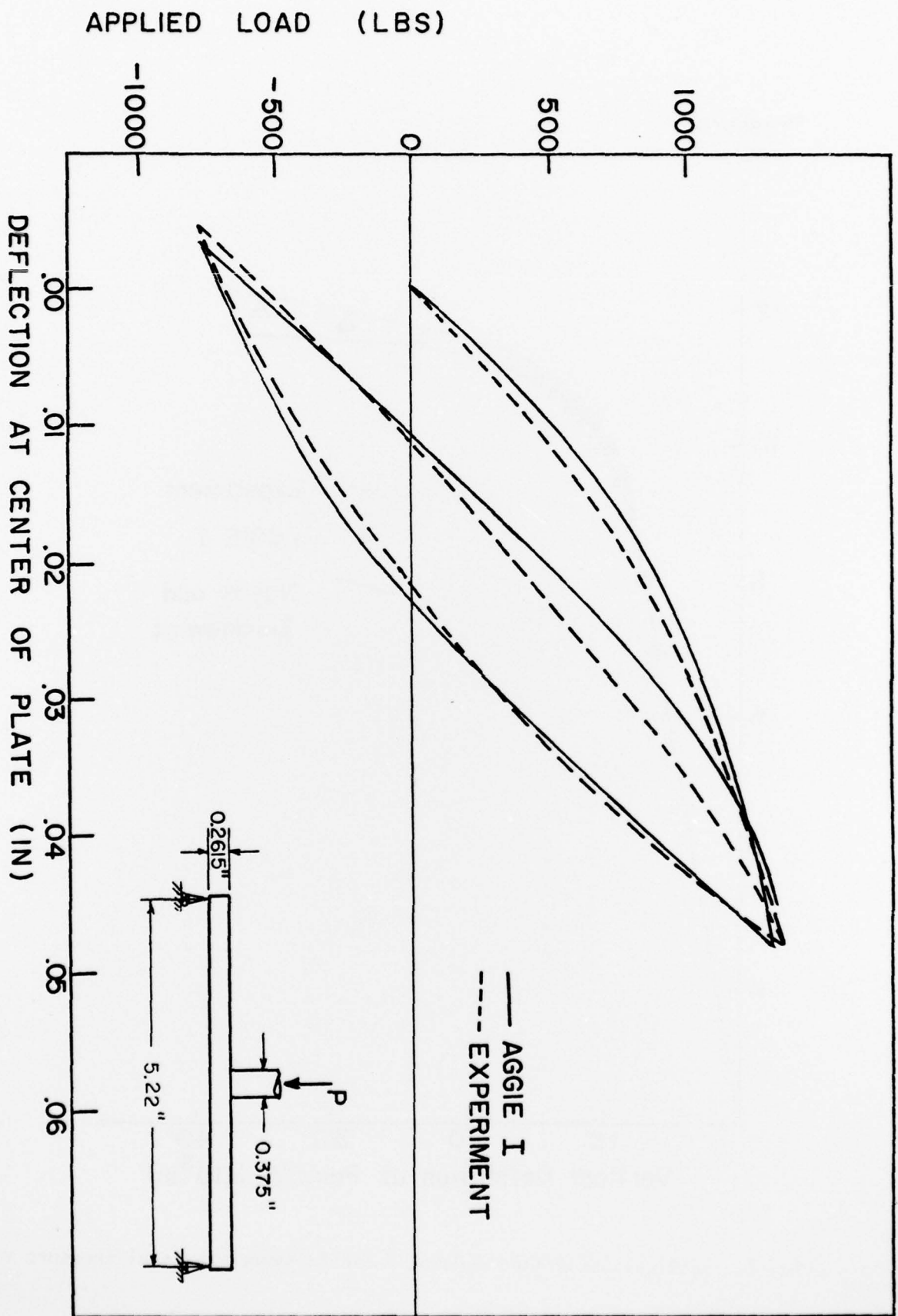


Fig. 8. Center Deflection of Circular Plate Subjected to Cyclic Punch Load - Combined Kinematic - Isotropic Hardening Model with $\gamma = 0.5$.

AGGIE I PROGRAM USERS

The initial version of the AGGIE I program as described herein has been released to a number of organizations during the past year. These include:

Office of Naval Research

NASA

Chrysler Corporation, Detroit, MI

Firestone Tire & Rubber Co., Akron, OH

Engineering Mechanics Research Corp., Troy, MI

TRW, Redondo Beach, CA

Universal Analytics, Playa Del Ray, CA

Wayne State Univ., Detroit, MI

Kaman Sciences Corp., Colorado Springs, CO

Requests for the program have also been received from:

Association of American Railroads, Chicago, IL

Diesel Allison, Indianapolis, IN

B.F. Goodrich, Akron, OH

CURRENT WORK

Work is currently underway which will enhance the analysis capability of the AGGIE I computer program. Specific improvements include:

- General shell element
- Thermal loading
- Material properties which are functions of temperature
- Creep material models

The general shell element is basically a doubly curved 4 to 8 node isoparametric element with 5 or 6 degrees-of-freedom per node (3 displacements and either 2 or 3 rotations). The element has been developed and verified for the linear, small strain case. Work is currently underway to adapt the element for large strain and nonlinear material behavior.

The thermal loading capability has been programmed into AGGIE I for the small and large strain case but has not yet been thoroughly checked out. The program allows for the input of temperatures at nodal points. For the case where material properties vary with temperature, the necessary modifications to the equilibrium equations have been developed theoretically. Programming of these modifications should begin in September.

Considerable work has been done in studying creep material models. An incremental, equation-of-state creep model which utilizes either a user-specified creep strain law or user-input discrete creep strain-stress-temperature data has been programmed for AGGIE and evaluated for the two dimensional elements. This model will be extended to the

three-dimensional elements as time permits.* At the present time, an incremental form of a nonlinear viscoelastic model is being developed and checked for the uniaxial case. The model is being programmed in such a way that we should be able to degenerate or modify it to include linear viscoelastic theory, Valanis theory, Krempl's theory, and Rashid's theory. This will allow us to evaluate the different approaches in a very fundamental basic way. Both numerical and experimental comparison studies will be done. We intend to investigate ways of including elastic-plastic-creep coupling and rate effects into the viscoelastic model (this may be possible through the shift factor and reduced time parameters). After this fundamental research has been finished, we will then incorporate the viscoelastic model into the AGGIE program.

ACKNOWLEDGEMENT

The research reported herein is being supported by the Office of Naval Research under Grant N00014-76-C-0150.

*Preliminary results using this model are given in Ref. 8.

REFERENCES

1. Bathe, K. J., Wilson, E. L., and Idling, R. H., "NONSAP: A Structural Analysis Program for Static and Dynamic Response of Nonlinear for Static and Dynamic Response of Nonlinear Systems," Report No. UC SESM 74-3, Structural Engineering Laboratory, University of California, Berkeley, CA, Feb. 1974.
2. Bathe, K. J., Ozdemir, H., and Wilson, E. L., "Static and Dynamic Geometric and Material Nonlinear Analysis," Report No. UC SESM 74-4, Structural Engineering Laboratory, University of California, Berkeley, CA, Feb. 1974.
3. Hunsaker, B., "The Application of Combined Kinematic-Isotropic and the Mechanical Sublayer Model to Small Strain Inelastic Structural Analysis by the Finite Element Method," Ph.D. Dissertation, Texas A&M University, Aug. 1976.
4. Takamatsu, T., "Nonlinear Finite Element Analysis of Incompressible Hyperelastic Materials Using Symmetric Stiffness Matrix," Ph.D. Dissertation, Texas A&M University, Dec. 1976.
5. Haisler, W. E., "AGGIE I - A finite Element Program for Nonlinear Structural Analysis," Report TEES-3275-77-1, Aerospace Engineering Department, Texas A&M University, June 1977.
6. Mondkar, D. P. and Powell, G. H., "Towrads Optimal In-Core Equation Solving," Computers & Structures, Vol. 4, 1974, pp. 531-548.
7. Mondkar, D. P. and Powell, G. H., "Large Capacity Equation Solver for Structural Analysis," Computers & Structures, Vol. 4, 1974, pp. 699-728.
8. Haisler, W. E. and Sanders, D. R., "Elastic-Plastic-Creep- Large Strain Analysis at Elevated Temperatures by the Finite Element Method," to be presented at the Symposium on Future Trends in Computerized Structural Analysis and Synthesis, Washington, D.C., Oct. 30 - Nov. 1, 1978.

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER 3275-78-2	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) Status Report of AGGIE I Computer Program		5. TYPE OF REPORT & PERIOD COVERED Interim
		6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) Walter E. Haisler		8. CONTRACT OR GRANT NUMBER(s) N00014-76-C-0150
9. PERFORMING ORGANIZATION NAME AND ADDRESS Department of Aerospace Engineering Texas A&M University College Station, TX 77843		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS NR 064-534
11. CONTROLLING OFFICE NAME AND ADDRESS Office of Naval Research Department of the Navy Arlington, Virginia 22217		12. REPORT DATE Unclassified
		13. NUMBER OF PAGES 26
14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office)		15. SECURITY CLASS. (of this report)
		15a. DECLASSIFICATION/DOWNGRADING SCHEDULE
16. DISTRIBUTION STATEMENT (of this Report) Unlimited <div style="border: 1px solid black; padding: 5px; display: inline-block;">DISTRIBUTION STATEMENT A Approved for public release; Distribution Unlimited</div>		
17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)		
18. SUPPLEMENTARY NOTES		
19. KEY WORDS (Continue on reverse side if necessary and identify by block number) Finite element computer program Structural Analysis Large Strain Plasticity		
20. ABSTRACT (Continue on reverse side if necessary and identify by block number) A status report of the AGGIE I structural analysis program is presented. AGGIE I is a computer program for predicting the linear and nonlinear, static and dynamic structural response of two- and three-dimensional continuum solids. The program is based on isoparametric finite elements and allows for two-dimensional plane stress, plane strain and axisymmetric analyses and general three-dimensional analyses. Large strain kinematics is based on the total Lagrangian formulation. Materially nonlinear models include several elastic- (cont.)		

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE(When Data Entered)

20. (Cont.)

plastic, work-hardening models as well as an incompressible Mooney-Rivlin model.

Included in this status report are several structural problems which have been investigated to verify the program. A listing of current program users is given. Also included is a summary of ongoing work which, when completed, will result in several improvements of the AGGIE I program.

A

1473B

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE(When Data Entered)